NSWCCD-50-TR-2010//028 Computational Analysis of Reagent Mixing in Ballast Water Tanks

Naval Surface Warfare Center
Carderock Division
West Bethesda, MD 20817-5700

NSWCCD-50-TR-2010/028 May 2010
Hydromechanics Department Report

Computational Analysis of Reagent Mixing in Ballast Water Tanks

by

Wesley M. Wilson

Approved for Public Release: Distribution Unlimited.

REPORT DOCUMENTATION PAGE

Form Approved
OMB No. 0704-0188

1. REPORT DATE *(DD-MM-YYYY)*
30 – 05- 2010

2. REPORT TYPE
Final

3. DATES COVERED *(From-To)*
09/09 to 03/10

4. TITLE AND SUBTITLE
Computational Analysis of Reagent Mixing in Ballast Water

5a. CONTRACT NUMBER

5b. GRANT NUMBER

5c. PROGRAM ELEMENT NUMBER
0604300N

6. AUTHOR(S)
Wesley M. Wilson – NSWCCD

5d. PROJECT NUMBER

5e. TASK NUMBER

5f. WORK UNIT NUMBER
09-1-5700-144

7. PERFORMING ORGANIZATION NAME(S) AND ADDRESS(ES)
NAVAL SURFACE WARFARE CENTER CARDEROCK DIVISION
9500 MACARTHUR BLVD
WEST BETHESDA, MD 20817-5700

8. PREFORMING ORGANIZATION REPORT NUMBER
NSWCCD-50-TR-2010/028

9. SPONSORING / MONITORING AGENCY NAME(S) AND ADDRESS(ES)
NATIONAL PARKS OF LAKE SUPERIOR FOUNDATION (CAROL BRADY)
P.O. BOX 632
HOUGHTON, MI 49931

10. SPONSOR/MONITOR'S ACROYNM(S)

11. SPONSOR/MONITOR'S REPORT

12. DISTRIBUTION / AVAILABILITY STATEMENT
Approved for public release: distribution unlimited.

20100621257

13. SUPPLEMENTARY NOTES

14. ABSTRACT
Invasive species continue to be a significant threat to local aquatic ecosystems in the US Great Lakes regions and other places around the world. One of the major contributors to the spread of non-indigenous aquatic species is the introduction of organisms in discharged ballast water. This and other related efforts are aimed at trying to gain a better understanding of the flow behaviors in ballast tanks and the impact of tank structure on the effectiveness of treatment methods to reduce or eliminate aquatic organisms resident in ballast water. This project uses computational methods to predict the spread of a passive scalar quantity, which simulates the use of a biocide to mix in the ballast water. Simulations have been performed to investigate the influence of inflow rate and several mixing mechanisms on the effectiveness of this type of treatment methodology. Here the sole metric is how well the biocide mixes in the tank. This does not make any judgment concerning the actual influence on organism population. It is the hope that this type of analysis may help to advance the understanding of ballast water treatment technologies for current and future applications.

15. SUBJECT TERMS Ballast Water, invasive species, mixing, biocide, CFD

16. SECURITY CLASSIFICATION OF:			17. LIMITATION OF ABSTRACT	18. NUMBER OF PAGES	19a. NAME OF RESPONSIBLE PERSON
a. REPORT	b. ABSTRACT	c. THIS PAGE			Wesley Wilson
UNCLASS	UNCLASS	UNCLASS	SAR	23+iv	19b. TELEPHONE NUMBER *(include area code)* (301) 227-5407

Standard Form 298 (Rev. 8-98)
Prescribed by ANSI Std. Z39.18

This page intentionally left blank.

Contents

Figures

Tables

Abstract

Invasive species continue to be a significant threat to local aquatic ecosystems in the US Great Lakes regions and other places around the world. One of the major contributors to the spread of non-indigenous aquatic species is the introduction of organisms in discharged ballast water. This and other related efforts are aimed at trying to gain a better understanding of the flow behaviors in ballast tanks and the impact of tank structure on the effectiveness of treatment methods to reduce or eliminate aquatic organisms resident in ballast water. This project uses computational methods to predict the spread of a passive scalar quantity, which simulates the use of a biocide to mix in the ballast water. Simulations have been performed to investigate the influence of inflow rate and several mixing mechanisms on the effectiveness of this type of treatment methodology. Here the sole metric is how well the biocide mixes in the tank. This does not make any judgment concerning the actual influence on organism population. It is the hope that this type of analysis may help to advance the understanding of ballast water treatment technologies for current and future applications.

Administrative Information

This work was coordinated by the United States Geological Survey (USGS) Leetown Science Center, with support from the Lake Superior Foundation. The work described in this report was performed by the Computational Hydromechanics Division (Code 57) of the Hydromechanics Department at the Naval Surface Warfare Center, Carderock Division (NSWCCD). The work was performed as part of Work for Private Parties agreement WFPP-57-0581 with work unit 09-1-5700-144 starting in FY2009 and completing in FY2010.

Acknowledgments

This work was performed in collaboration with Dr. Barnaby Watten from USGS Leetown Science Center. His support is greatly appreciated. The support of the US Department of Defense High Performance Computing Modernization Program (MPCMP) that provided the computer resources at the AFRL DSRC is also greatly appreciated.

Introduction

Researchers at the Naval Surface Warfare Center – Carderock Division have been actively investigating certain aspects of invasive species in ballast water over the past several years. In particular, efforts have focused on computational predictions of ballast water exchange procedures in bulk carriers[1,2]. Computational fluid dynamics (CFD) methods have been applied to simulating a ballast water exchange event aboard a bulk carrier. One of the outcomes of this investigation was in determining the appropriate level of details about the internal tank structure necessary to provide good results. A picture of the full-scale ballast tank model is shown in Figure 1. A sample calculation from previous ballast water exchange simulations is shown in Figure 2. The figure details the impact of the internal tank structure.

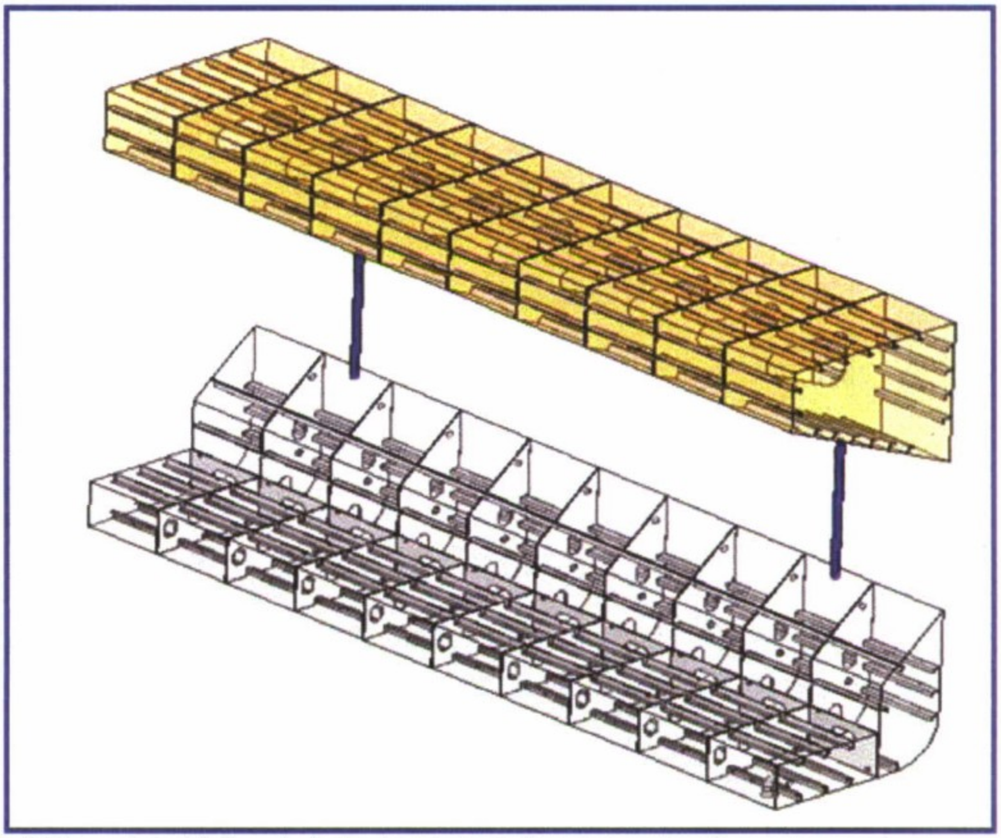

Figure 1: CAD model of full-scale ballast tank, including internal tank structure and connection pipes.

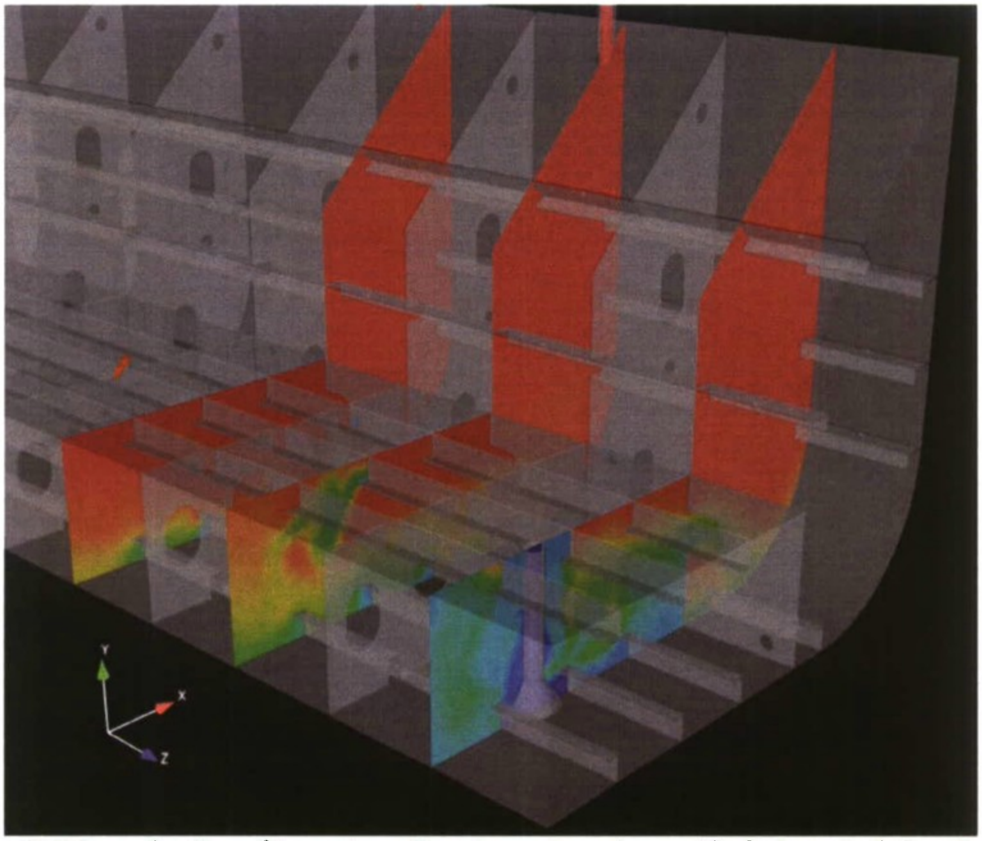

Figure 2: Volume fraction of incoming salt water contours in specified planes in full-scale ballast tank (red = pure resident fresh water, blue = incoming salt water).

Approach

The intent of this effort is to try to gain some understanding of how a treatment reagent might mix in a similar type of ballast tank using an air lift mixer. Here, the tank is a closed system (no inflow/exit of resident ballast water). A specified flow rate is then applied to one of the connection pipes that connect the upper wing tanks with the hopper side tanks (see Figure 1). This creates a circulating flow through the remaining tank bays that will potentially promote the mixing of the reagent. In this case, the reagent is simply treated as a passive scalar that is mixed throughout the tank by means of simple advection and diffusion. In order to assess the effectiveness of this type of mixing, a matrix of simulations has been performed that involve changes in volumetric flow rate, different injection methods, and potential uses of the air-lift mixing technique in situations where there may only be a single connection pipe between the upper wing tops and hopper side tanks. This matrix of simulations is given below.

Runs 1 and 2 will look at two different inflow rates for a batch method of introducing the reagent. The two flow rates are 1100 gal/min and 750 gal/min. These are typical values that might be used in a full-scale container ship application. In Run 3 and Run 4 a continuous feed injection method was examined for introducing the reagent. In one case the feed was introduced into an upward flow through one of the connection pipes, such that the reagent was first introduced into the upper wing tanks. In the other case, the feed was introduced into a downward flow through the second connection pipe, such that the reagent was first introduced into the double bottom tank compartments. For Run 5, only a single connection pipe between the upper wing tanks and hopper tanks was utilized. In order to generate a circulating flow in the tank an airlift eductor system was used that spans across the different tank bays. Here the water is drawn in from one bay and discharged in another, spatially distant, tank bay. More details concerning the simulation cases is given in later sections. The run matrix is summarized in Table 1.

Table 1. Simulation Suite: Full-Scale Tank Geometry

Run #	Flow rate	injection method
1	1100 gal/min	batch
2	750 gal/min	batch
3	1100 gal/min	continuous feed (upper ingress)
4	1100 gal/min	continuous feed (lower ingress)
5	1100 gal/min	single connection pipe w/eductor

Geometry Details

This section will provide some more details concerning the tank geometry that was used and the airlift eductor system that was used in Run 5. The full-scale tank model is a physical representation of one of a set of ballast tanks from a typical 35,000 dwt handysize bulk carrier. There are a total of ten rows of double bottom tank, hopper side tank, and topside tank bays (see Figure 1). The topside tank bays are connected to the hopper tank bays via two connection pipes. The tank floor openings and stiffeners were modeled based on tank drawings provided by Fednav International. In a typical ballast water exchange exercise, the incoming fluid is pumped into the tank via a single inlet bellmouth. The exiting fluid during a ballast water exchange would then be discharged through openings in the tank tap and allowed to flow overboard. In this present exercise, the inlets and exits were not used, and the tank was assumed to be a closed system. During a real application of the biocide reagent, the mixing might take place during a transit, at which point the tank inlets and exits would be closed.

For Run 5, an airlift eductor system was used. This is a technique that is currently being examined to improve the effectiveness of biocide mixing. More details concerning this approach will be given in a later section.

Computational Method

The CFD simulations are performed using the commercial viscous flow solver, Fluent, developed by Ansys, Inc. The code uses a cell-centered finite volume approach to solving the governing equations of mass and momentum. In addition, a transport equation is solved for the volume fraction of one of the fluids, with the algebraic constraint that the volume fraction of both fluids must sum to unity. The convection terms are discretized using a second order upwind method, while the diffusion terms are discretized using the second order accurate central differencing scheme. The turbulence closure used the realizable k-ε model. In order to solve for the physical and temporal distribution of the two fluids, the mixture multi-phase model in Fluent is used. The reagent properties are assumed to be similar to that of hydrated lime, and a fluid density of 1258.74 kg/m^3 is used. The dynamic viscosity is assumed to be equal to that of water, and the hydrated lime is assumed to be the secondary phase in the mixture model and treated as a passive scalar. The time derivative terms are discretized using the first order backward implicit scheme. The PISO pressure-velocity coupling method is used, and the discretized equations are solved using pointwise Gauss-Seidel iterations, and an algebraic multi-grid method accelerates the solution convergence.

Previous studies have focused on validation exercises related to simulations of ballast water exchange. Here model-scale experiments were used to provide data to validate the CFD simulation results for a 1/3-scale model of four bay compartments[1,2,3]. After having successfully validated the computational model, simulations were also extended to the full-scale tank geometry[3], which is currently being used in this project. No further validation exercises have been performed as part of this project, and it is assumed that the CFD model has been validated sufficiently to provide confidence in the present predictions.

The simulations were performed as fully time-accurate. This is necessary in order to model the physical time it takes for the mixing of the biocide to occur. The time step was initially rather small (approximately 1.0E-03 seconds) at the beginning of each simulation in order to allow for the initial transients to settle out. Then the time step was gradually increased to approximately 0.5 seconds. The time scale of the tank mixing is on the order of several hours, which requires tens of thousands of iterations to perform a complete mixing simulation. At the beginning of each simulation, the flow rate through the connection pipe was established through a "fan" boundary condition available in Fluent. This requires the application of a specified pressure drop across the boundary plane of the fan. This value was then iterated on until the appropriate volumetric flow rate was achieved. An example of the predicted vertical velocity is shown in Figure 3, which represents a cut plane that intersects the upstream vertical connection pipe.

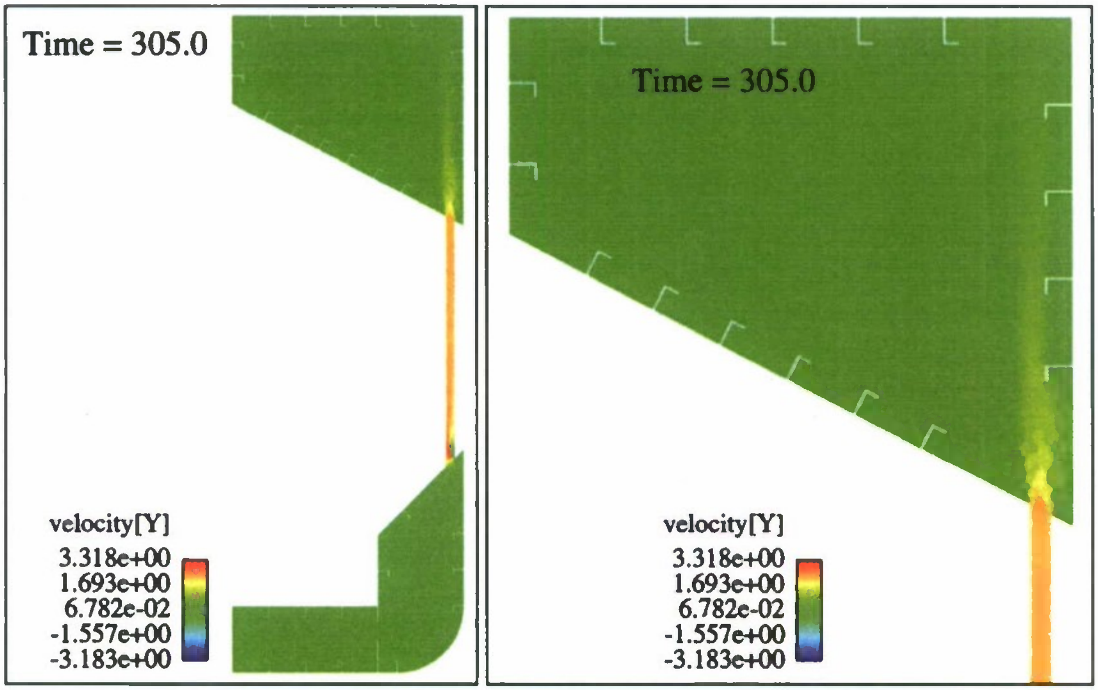

Figure 3. Vertical velocity contours in x-z plane through center of connection pipe, including enlarged view of upper wing tank.

After establishing the correct flow rate through the connection pipes the scalar representing the hydrated lime was then introduced into the tank, either in a batch mode in a single location, or as a continuous feed through the connection pipe. During the remaining simulation time, the transport of the scalar was tracked and monitored in each of the tank compartments. Specific monitors were established to record the total volume of scalar that was present in each tank compartment as a function of time.

In order to improve the efficiency of the simulations, the previously generated tetrahedral mesh (used in a prior ballast water study[1]) was converted to polyhedra, which are constructed from an arbitrary number of sides. Computational meshes constructed from polyhedra have several advantages, including reduced computational cell count and greatly improved mesh quality through reduced cell skewness. Both of these attributes have a significant, positive effect on solution convergence. For this case, the tetrahedral mesh contained approximately 7.5 million cells, while the converted polyhedra mesh contained approximately 2.4 million cells. A comparison of the mesh applied to the wall surfaces is shown in Figure 4. This is a detailed view of the upper corner section of one of the upper wing tank compartments. The surface mesh is shown on the dividing bulkhead and stringers. The view shows the large opening through the dividing bulkhead.

The CFD simulations were performed using computational resources available at the Air Force Research Laboratory (AFRL) DoD Supercomputing Resource Center (DSRC).

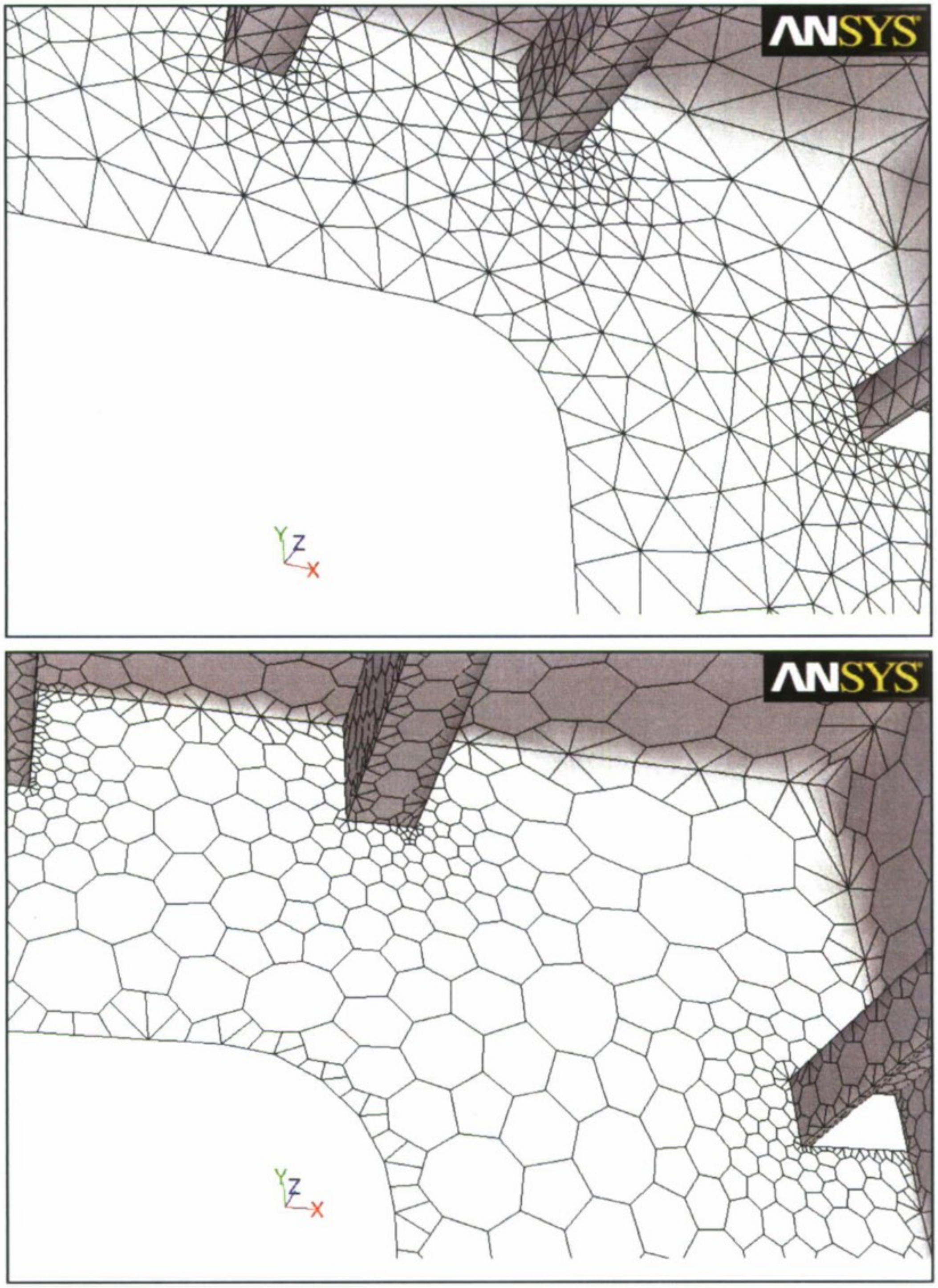

Figure 4. Comparison of mesh details for tetrahedral vs. polyhedra meshes (view detail from upper corner of upper wing tank compartment).

Results

In this section the results of the computational run matrix is presented. The results will first be presented for each case. Following this, some comparisons of the different injection methods is provided.

Case 1: Batch Injection (Q=1100 gpm)

The first case that was simulated was for a batch injection of the scalar at a single location in the tank, with a volumetric flow rate of 1100 gal/min through the connection pipe. The location where the scalar was introduced was near the top of the tank in upper wing compartment 3, next to one of the stringers. The initial volume of scalar that was introduced was 3.97E-05 m^3, which corresponds to a volume concentration of approximately 60 parts per billion (ppb). The compartment numbering convention is shown in Figure 5.

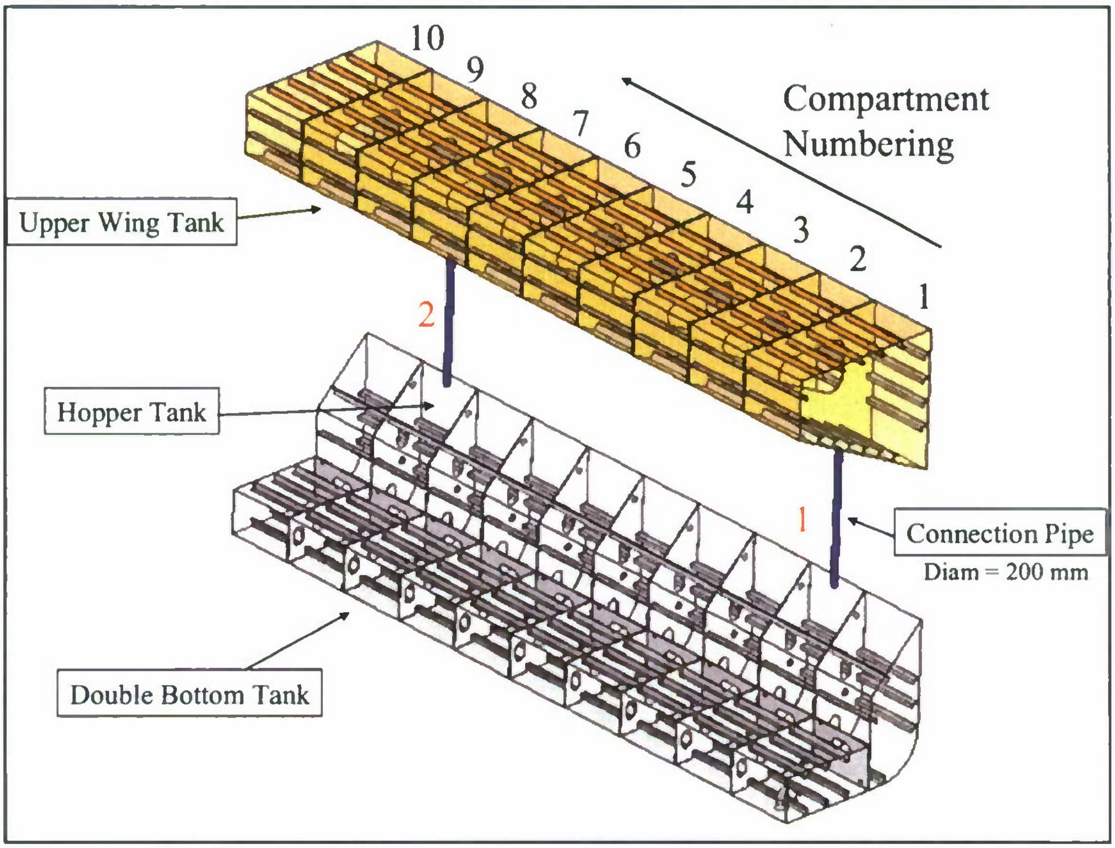

Figure 5. Compartment numbering convention.

An example of the predicted volume fraction of the hydrated lime shortly after it was introduced into the tank is shown in Figure 6. As shown in the figure, after an elapsed flow time of approximately 5 minutes, the lime has begun to be advected away from the seed site, and some is also falling downward through the upper wing tank because the introduced fluid is denser than the resident water.

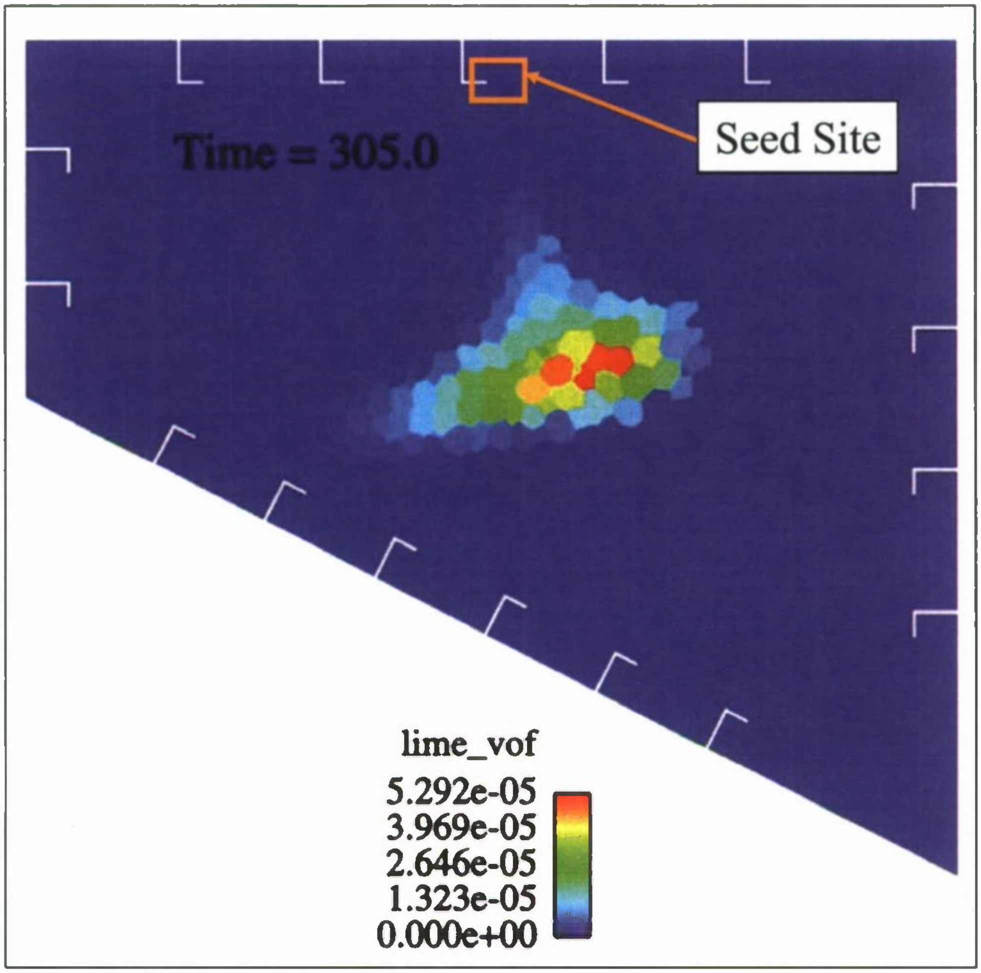

Figure 6. Predicted contours of volume fraction of hydrated lime in upper wing tank compartment 3 at Time = 305 secs.

The CFD simulations were performed in a fully transient manner, and tracked the evolution of the hydrated lime distribution within the tank as a function of time. A series of solution monitors were established to record the integrated volume of the hydrated lime in each of the tank compartments as a function of the elapsed flow time. It is assumed that after a sufficiently long time the concentration of the hydrated lime will reach an equilibrium state. In order to measure the effectiveness of mixing the scalar quantity in the tank, a metric has been termed the Compartment Equilibrium Index (CEI). This metric provides a measure of the predicted percentage of each compartment volume that is occupied by the hydrated lime scalar at any given time divided by the expected volume percentage of the lime at equilibrium for each compartment. The reason that this is defined per compartment is because the total available volume of each compartment is not always the same because, for example, the upper wing tanks are larger than the double bottom tanks. A comparison of the CEI values at each 60 minute interval of elapsed simulation time is shown in Figure 7.

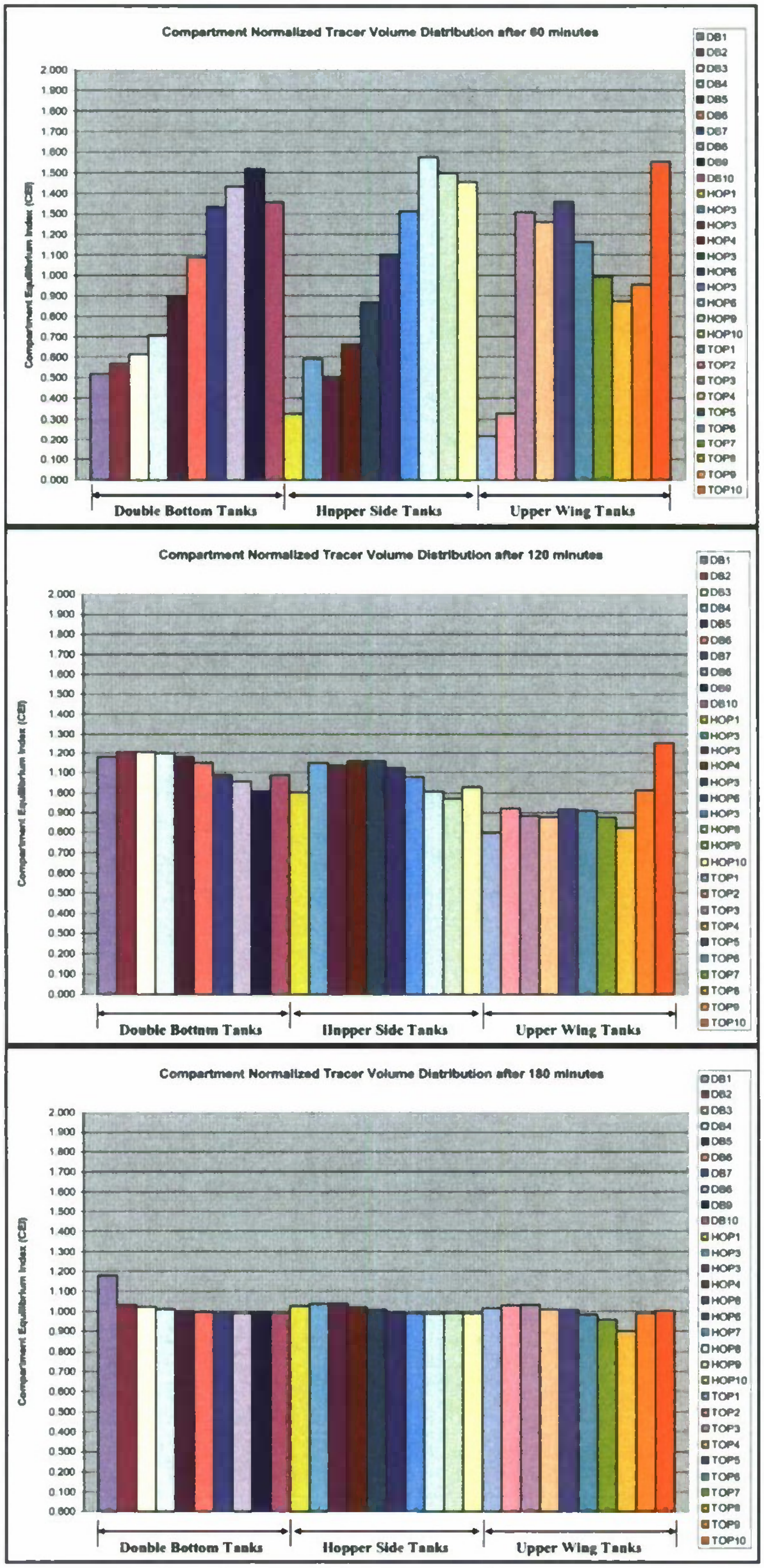

Figure 7. Compartment Equilibrium Index (CEI) values at hourly intervals (Run Case 1)

As shown in Figure 7 as the solution evolves over time, the CEI values for each compartment move closer and closer to a value of 1.0, which indicates perfect equilibrium.

Case 2: Batch Injection (Q=750 gpm)

The second case in the run matrix examined how changing the flow rate might effect the effectiveness of mixing the hydrated lime throughout the tank. Of coarse, it is anticipated that with a decreased flow rate, the mixing effectiveness would be decreased.

The predicted compartment equilibrium index (CEI) values at hourly intervals is shown in Figure 8. In this case the simulations were extended for an additional 120 minutes due to the fact that the mixing of the hydrated lime was taking much longer. Even after 300 minutes, the scalar concentration is not nearly as close to equilibrium (CEI = 1.0) as in the previous case where the flow rate was higher.

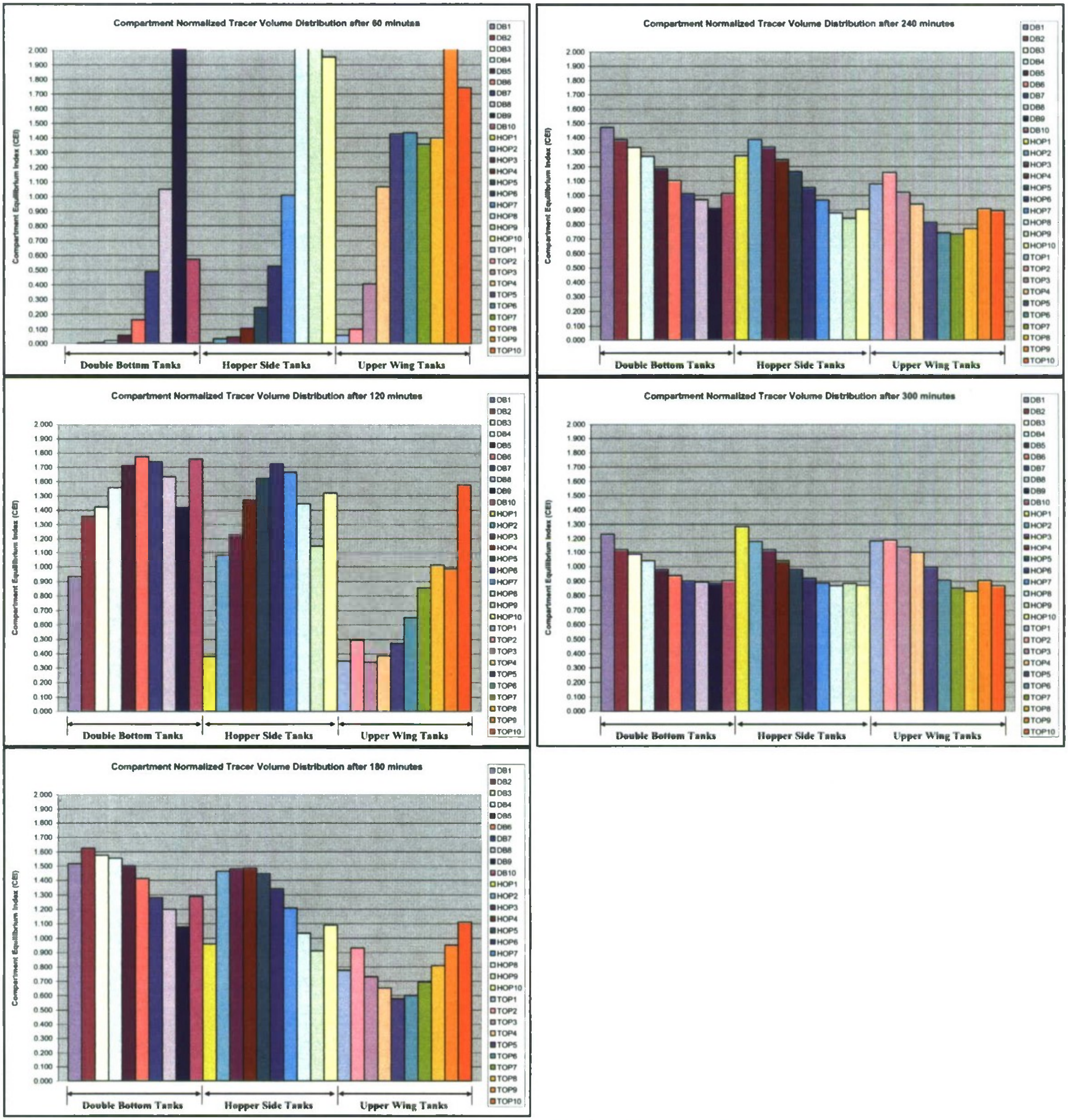

Figure 8. Compartment Equilibrium Index (CEI) values at hourly intervals (Run Case 2)

Case 3: Continuous Upward Feed (Q=1100 gpm)

The next two run cases examined the concept of using a continuous feed of the hydrated lime directly into the connection pipe. The feed was performed slowly, such that the process took approximately 60 minutes, and arrived at the same total volume of scalar introduced into the tank. For Run Case 3 the feed was injected into connection pipe 1 (see Figure 5), where the flow was directed vertically upward into the upper wing tanks.

The predicted CEI values at hourly intervals is shown in Figure 9. A comparison of Figure 9 and Figure 7 shows that the hydrated lime concentration seems to be approaching equilibrium more quickly for the continuous feed case than for the batch introduction case. Also, the increase in the lime concentration as a function of time is more evenly distributed throughout the tank. This type of reagent introduction could be achieved during a filling or ballast water exchange procedure and would help improve the ability to mix the biocide throughout the entire ballast water tank in an efficient manner.

Case 4: Continuous Downward Feed (Q=1100 gpm)

For the second continuous feed case, the scalar was introduced slowly into connection pipe 2, where the flow was directed downward into the double bottom tanks. Again, the feed required 60 minutes to introduce the equivalent volume of hydrated lime as was used in the batch introduction Run Case 1. When examining the continuous feed options it became obvious that the evolution of the scalar concentration could be significantly different depending on whether the flow was directed into the upper wing tanks, which are larger and more open tanks, with minimal tank structure or directed into the double bottom tanks, which are much smaller and shorter and have significant tank structure to impede mixing.

The predicted CEI values at hourly intervals are shown in Figure 10. Because the feed is into the lower tanks, there is a larger disparity in the concentration in the double bottom tanks and hopper side tanks as opposed to the upper wing tanks. This causes a very different evolution of the hydrated lime concentration because there is much more intervening structure in the double bottom tanks. The upper wing tanks have large openings in the dividing bulkheads, which allow for much more freedom for the lime to be advected downstream. The double bottom tanks, on the other hand, have much more tightly closed compartment spaces, with dividing bulkheads that contain only personnel access manholes and smaller limberholes. By comparing Figure 10 with Figure 9, we can see that Run Case 4 more slowly approaches an equilibrium state.

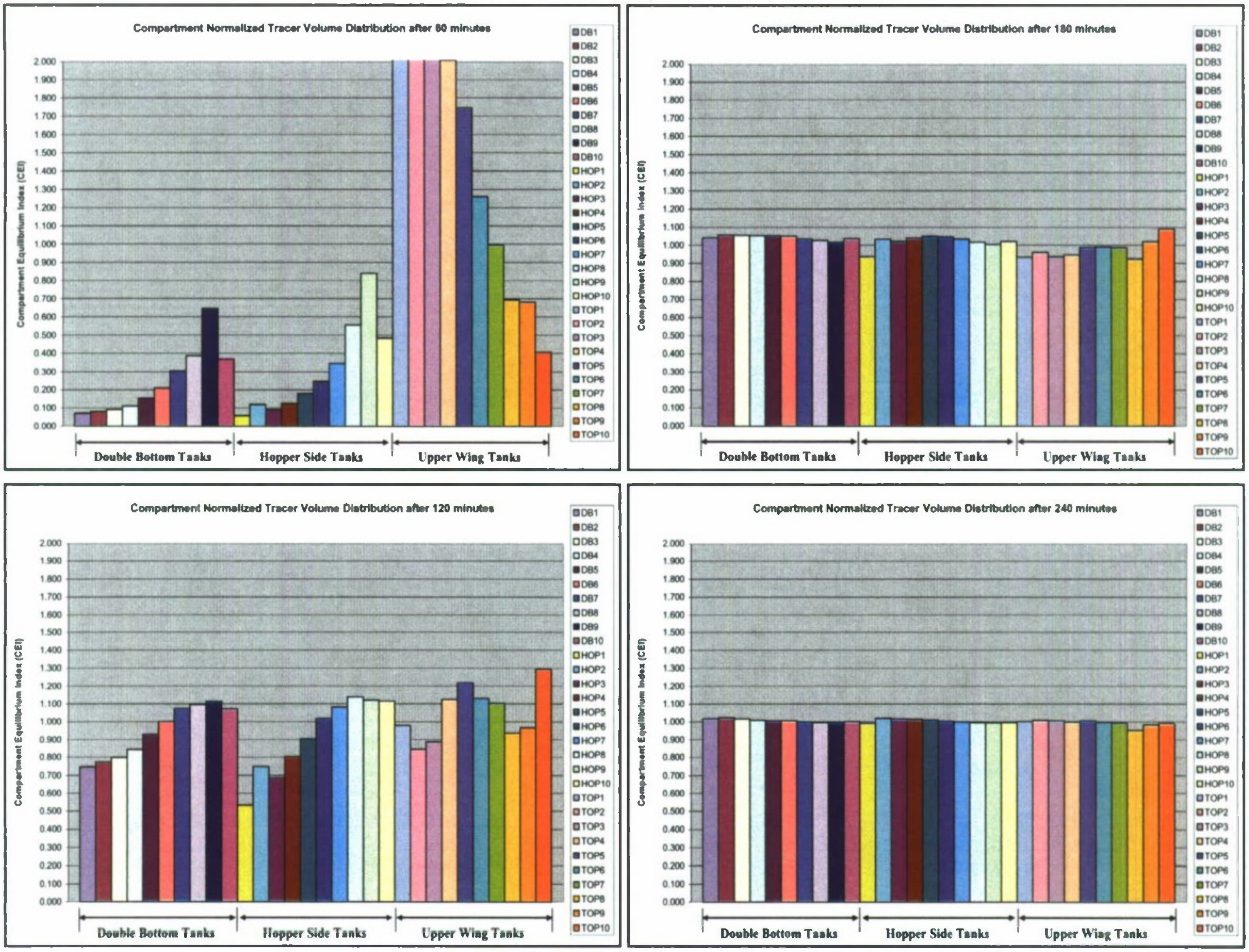

Figure 9. Compartment Equilibrium Index (CEI) values at hourly intervals (Run Case 3)

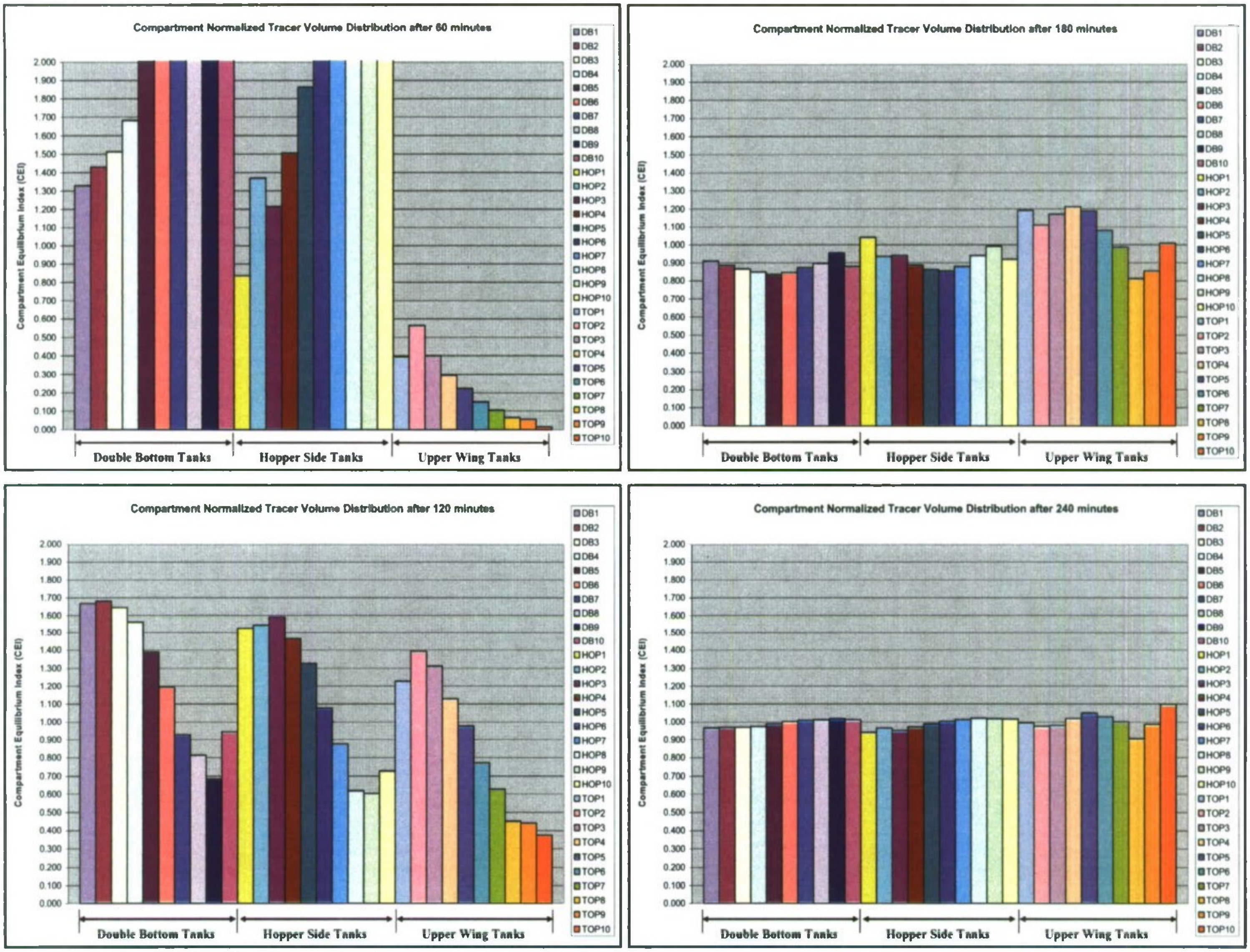

Figure 10. Compartment Equilibrium Index (CEI) values at hourly intervals (Run Case 4)

Another useful metric is to take the average CEI value across all of the compartments. As this value approaches 1.0 then the scalar concentration of the hydrated lime is approaching equilibrium. The average CEI gives a single number that can be used to compare the different simulation results. This value has been computed at every 60 seconds. A comparison of the change in the average CEI as a function of time is shown in Figure 11. As shown here, Run Case 2, which was at a lower flow rate, appears to be approaching equilibrium much more slowly than the other cases. We can also see that by comparing the two continuous feed cases, Feed1 (scalar introduced upward in connection pipe 1 into upper wing tanks) performs better than Feed2 (introduced downward in connection pipe 2 into double bottom tanks). Finally, by comparing Run Case 1 and Run Case 3, the Feed1 option appears to approach an average CEI value very close to one sooner than in the batch introduction case. This is again as expected, as the mixing of the scalar is increased by using the moving fluid in the connection pipe to more easily transport the scalar.

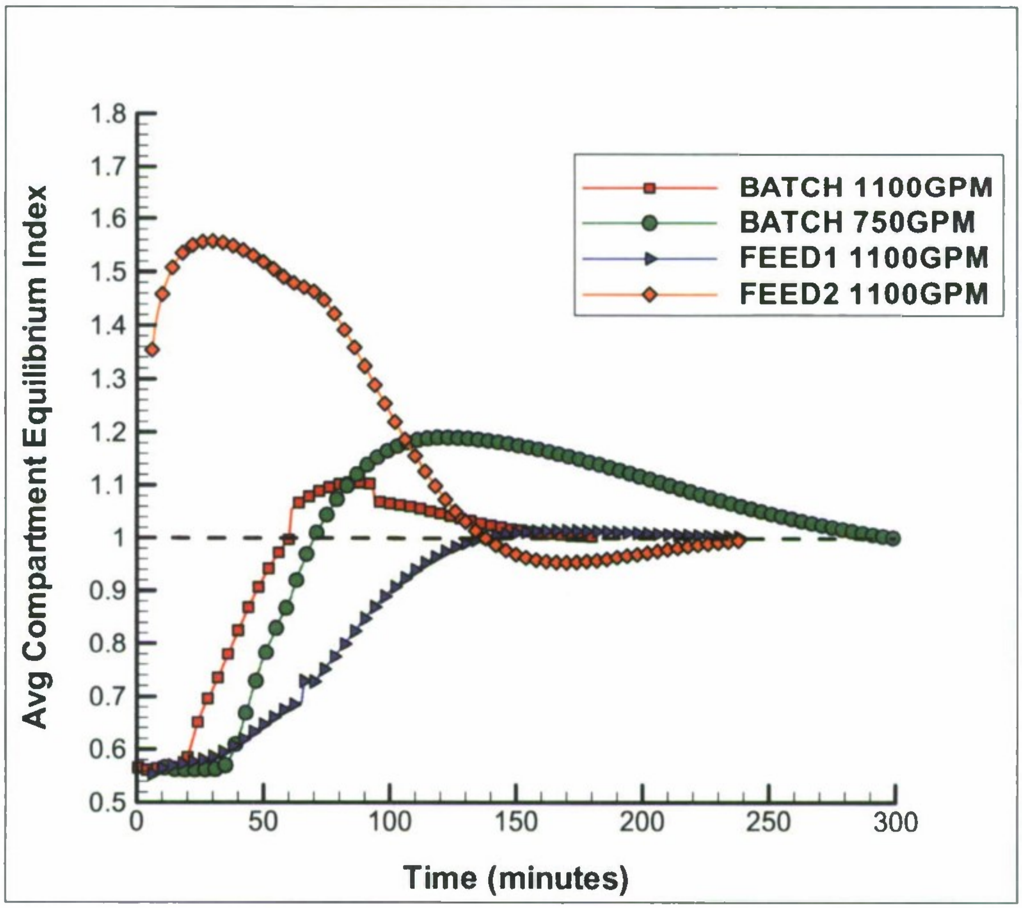

Figure 11. Comparison of Predicted Average Compartment Equilibrium Index (Run Cases 1-4)

Case 5: Airlift Eductor (Q=1100 gpm)

This section will provide details concerning Run Case 5, in which a technique is explored to improve the mixing effectiveness through the use of an airlift eductor. Essentially, the concept is to utilize the introduction of air bubbles into one section of a long pipe that will induce a pumping action through the pipe and draw fluid from one section of the tank to another section. This arrangement is illustrated in Figure 12.

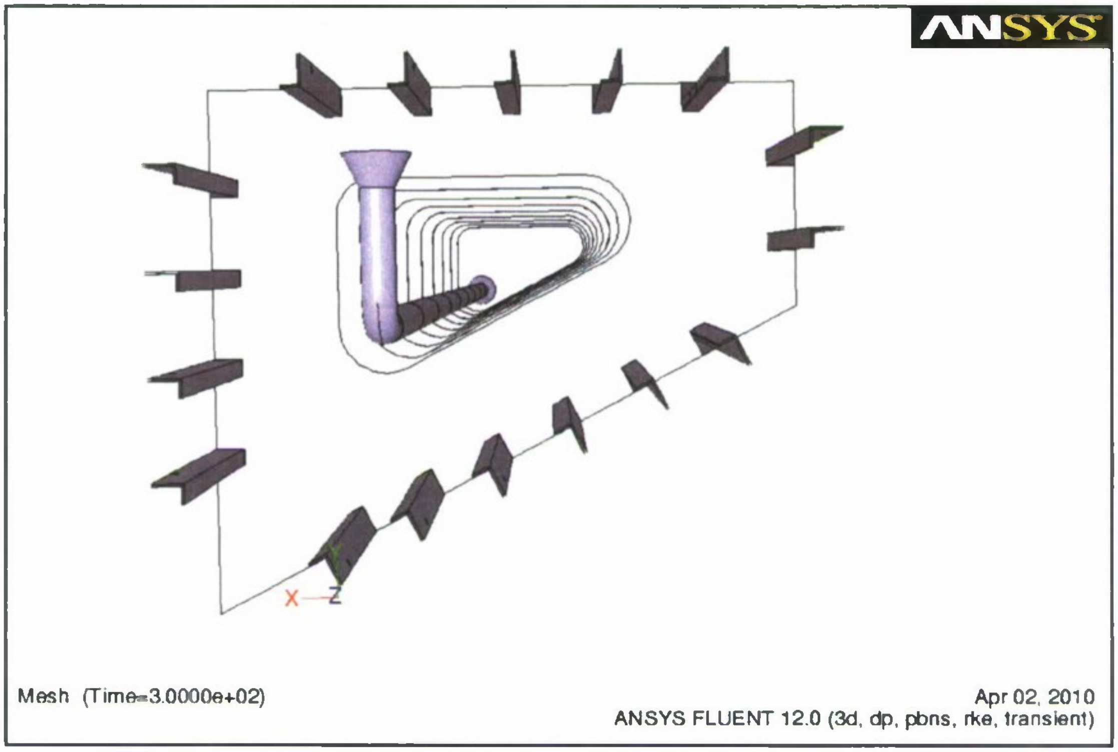

Figure 12. View of geometry details for Run Case 5 (view looking from upper wing compartment 10 towards upper wing compartment 1).

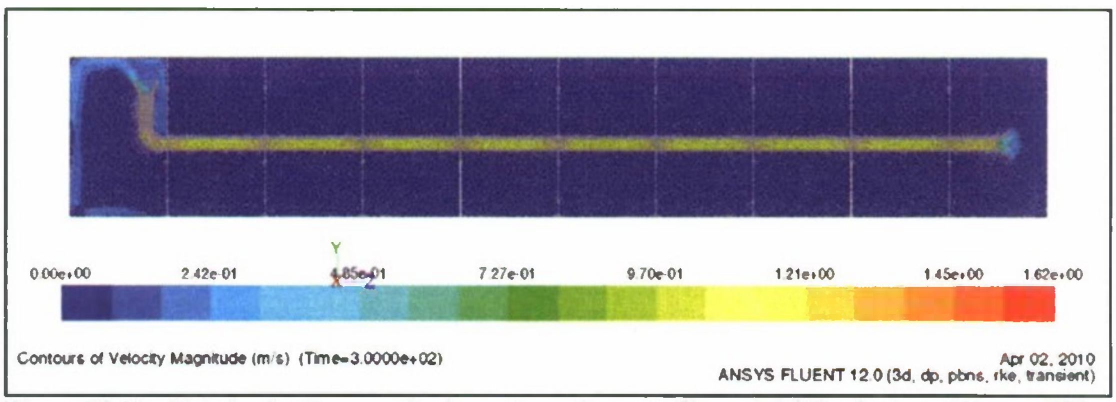

Figure 13. Predicted velocity magnitude contours in centerline of airlift eductor at Time = 300 seconds.

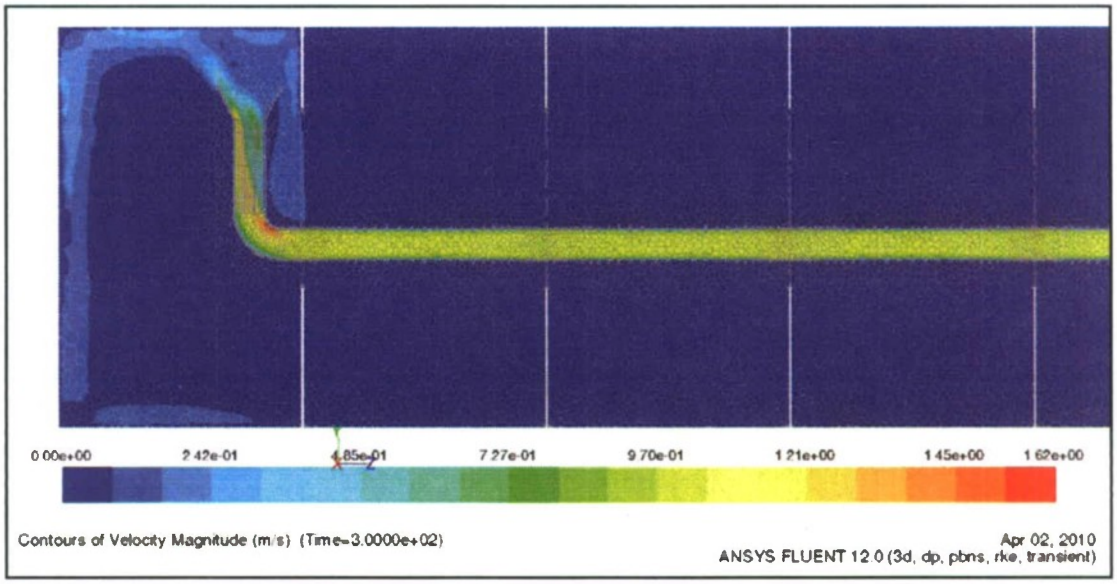

Figure 14. Predicted velocity magnitude contours in centerline of airlift eductor at Time = 300 seconds (zoomed view of exit bellmouth).

Contours of the predicted velocity magnitude are shown in Figure 13 along the centerline of the airlift eductor. Figure 14 shows a zoomed in view of the area near the exit bellmouth. As shown in these figures, the airlift eductor concept draws fluid from one are of the tank, and accelerates the fluid and deposits it in a geometrically distant area of the tank.

The predicted compartment equilibrium index values for the upper wing tanks are shown in Figure 15 at hourly intervals. Again, as the time elapses the concentration of the lime approaches an equilibrium value in each compartment. What is observed here, however, is that for Run Case 5 in examining the airlift pump concept, the equilibrium appears to be approached rather slowly. It was anticipated that this concept would help to promote a much quicker mixing in the tank, but the CFD predictions do not seem to agree. A model test evaluation of this concept is currently underway at the USGS Leetown Science Center facility to examine how effective it is at improving the mixing in the tank. Based on the results of that study, the CFD predictions may need to be revisited if it turns out that they do not agree with the model test results.

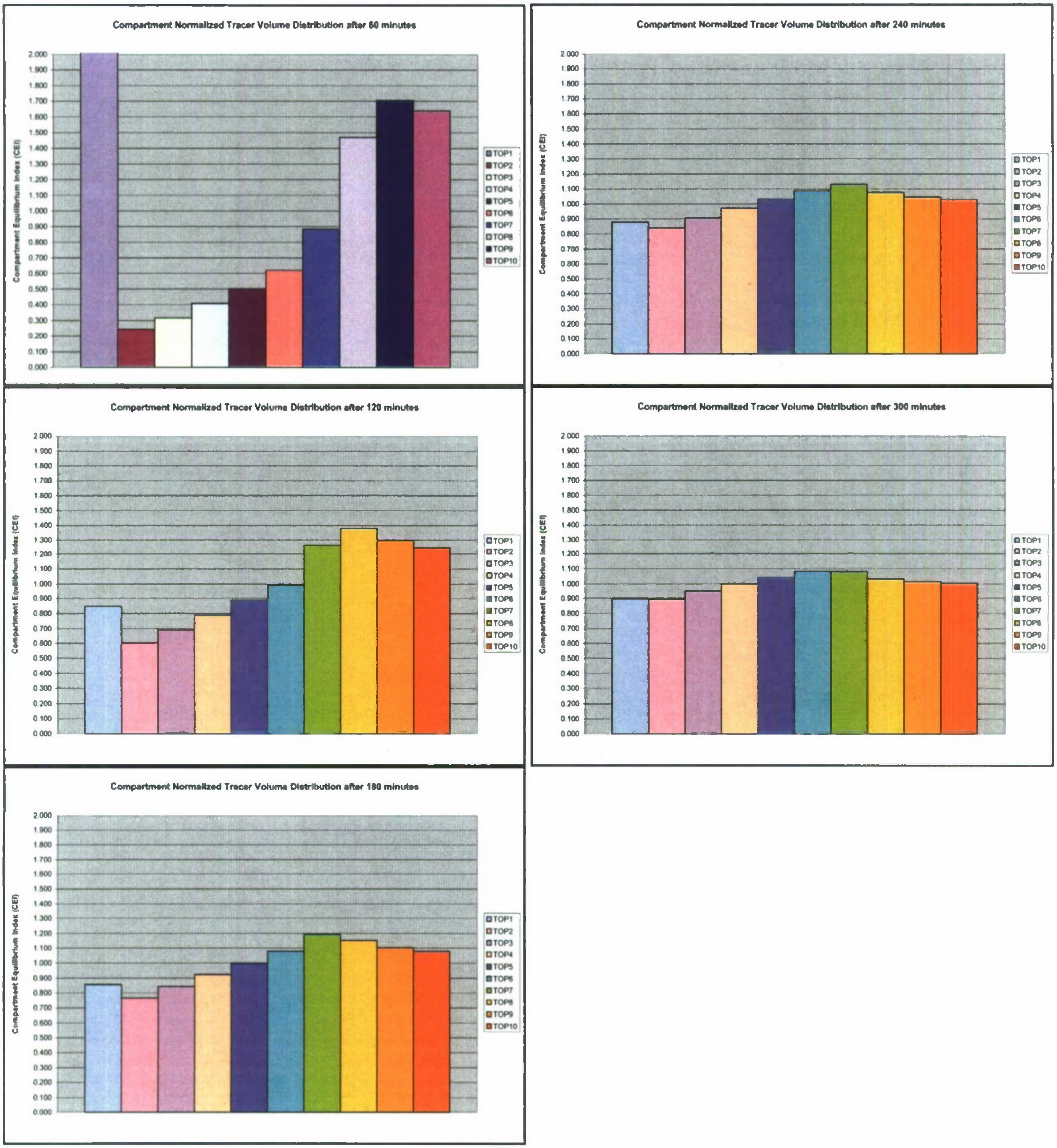

Figure 15. Compartment Equilibrium Index (CEI) values at hourly intervals (Run Case 5)

This page intentionally left blank.

Summary

Non-indigenous species is a growing problem that affects many of the worlds ecosystems. Ballast water that is carried in typical commercial shipping vessels has been identified as a leading vector for the transport and spread of invasive aquatic species. This report summarizes efforts to examine mixing in ballast tanks, with specific interest in improving mixing phenomena for treatment methods like biocides. Analyses have been performed using computational fluid dynamics (CFD) to examine the influence of inflow rate and introduction method on the effectiveness of mixing a passive scalar quantity through a model of a bulk carrier ballast tank system. Here the passive scalar is representative of a small quantity of hydrated lime that might be used as a biocide agent to kill off any organisms present in the tanks.

The results of the CFD simulations have been represented by the compartment equilibrium index (CEI), which is a measure of how quickly the scalar concentration of lime approaches the expected equilibrium concentration value. A comparison of this metric across the different run cases would seem to indicate that a higher inflow rate has a positive influence on the mixing in the tank. Also, introducing the lime through a continuous feed mechanism, as opposed to a batch introduction of the same quantity of lime, has a positive influence on approaching equilibrium more quickly.

Finally, a proposed method for increasing the mixing effectiveness by use of an airlift pumping mechanism has been examined. It was expected that this might have a positive benefit in reaching equilibrium more quickly than standard methods; however, the results of the CFD simulations do not agree with this supposition. There is currently a model test examination of this concept underway.

The knowledge gained from this and similar studies will advance the effectiveness of current and future ballast water treatment methods to reduce the spread of invasive aquatic species in ballast water.

This page intentionally left blank.

References

1. Wilson W., Chang P., Verosto S., Atsavaprance P., Reid D.F., Jenkins P.T.(2006), "Computational and experimental analysis of ballast water exchange," NAVAL ENGINEERS JOURNAL 118 (3): pp. 25-36.

2. Reid, D., Verosto, S., Chang, P., Wilson, W., Atsavaprance, P. and Jenkins, P. (2005) "Modeling of Ballast Water Mixing and Flow Dynamics to Understand Ballast Water Exchange," ENSUS 2005, Marine Science and Technology for Environmental Sustainability, Newcastle, UK.

3. Wilson W., Chang P., Verosto S., Atsavaprance P., Reid D.F., and Capt. Phillip T. Jenkins (2006), "Computational and experimental analysis of ballast water exchange," 2006 ASNE Marine Environmental Engineering Technology Symposium (MEETS), Arlington, VA, January 23-24.

REPORT DISTRIBUTION

No. of Copies			
Print	**PDF**	**Office**	**Individual**
1	1	USGS Leetown Science Center	Dr. Barnaby Watten
1		DTIC	
		NSWCCD Code	**Individual**
	1	3452	Library
1		5060	D. Walden
	1	5700	W. Wilson, J. Gorski, P. Chang
	1	5800	R. Hurwitz

Total No. of Copies	
Print	**PDF**
3	6